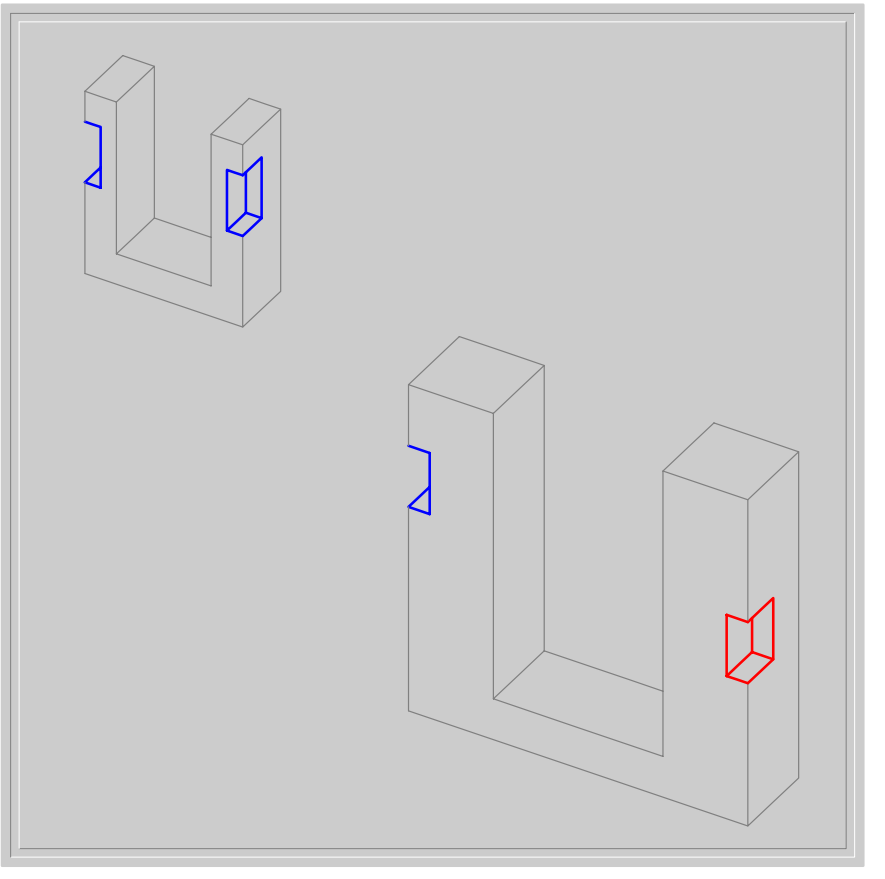


Adding Features with Associativity

I-DEAS® Tutorials: Fundamental Skills

Learn how to:

- create relationships using *Focus*
- create relationships by matching dimensions
- use simple equations
- add and delete relationships

Before you begin...

Prerequisite tutorials:

1. Getting Started (I-DEAS™ Multimedia Training)

   –or–

   Introducing the I-DEAS Interface,
   –and–
   Quick Tips to Using I-DEAS
2. Creating Parts
3. Sketching and Constraining
4. Dimensioning
5. Building Sections
6. Using Sketch Planes and Understanding Sketch Pads
7. Extruding and Revolving Features

Setting your defaults

What:

Before continuing, set the following default options for this tutorial.

How:

Options
Preferences...

Preferences form


Modeler/Assembly...

Modeler/Assembly Preferences form

3D part VGX (off)
Drag for extrude creation (off)
OK

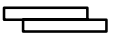
OK

Why:

With these recommended settings, the tutorial steps will work as documented. Other settings may cause minor changes in the required steps.

For more information, use *Help, on Context* and then pick the specific item of interest.

If you didn't start I-DEAS with a new (empty) model file, open a new one now and give it a unique name.


File
Open

Open Model File form


Model File name: any unique name

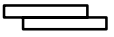

OK

Make sure you're in the following application and task:


Design, *Manufacturing*, or *Simulation*

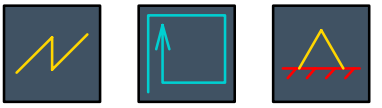

Modeler

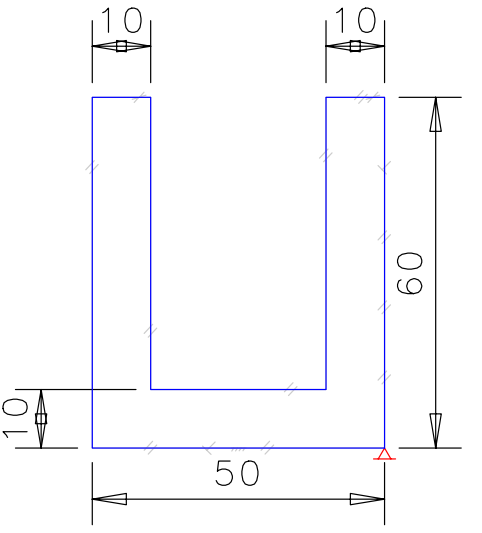
Set your units to mm.


Options
Units

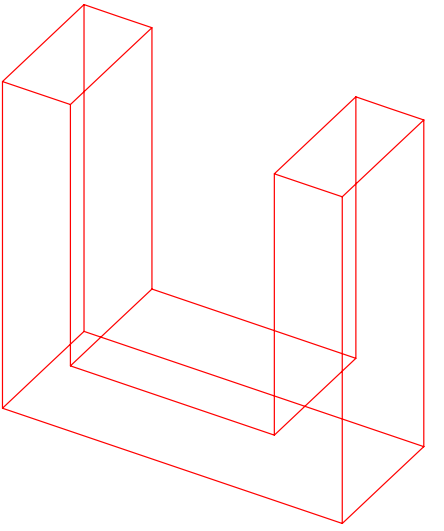

mm (*milli newton*)

Step 1. Create the wireframe to the dimensions shown. Then anchor the bottom right point.

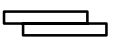




Step 2. Extrude to 20mm.



Save your model file.



File
Save

Warning!
If you are prompted by I-DEAS to save your model file, respond:

No

Save only when the tutorial instructions tell you to—not when I-DEAS prompts for a save.

If you make a mistake at any time between saves and can't recover, you can reopen your model file to the last save and start over from that point.

Hint
To reopen your model file to the previous save, press Control-z.

Create relationships using *Focus* 1 of 7

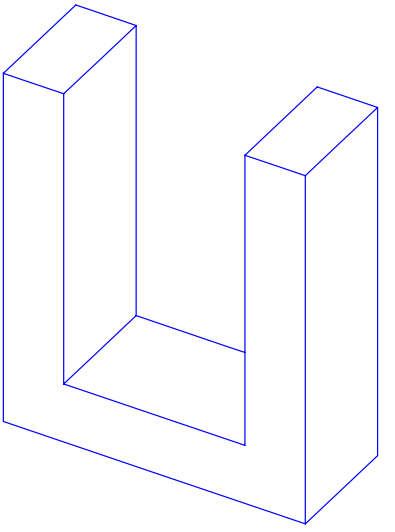
You can define a relationship between features by sketching wireframe with the *Focus* option (right mouse button menu).

Focus projects a point from another feature to the 2D sketch plane you are working on. If this projected point is used in the sketch, a positional relationship is created. Use this technique to create features that maintain a positional relationship to each other, even if you modify the dimensions of the part.

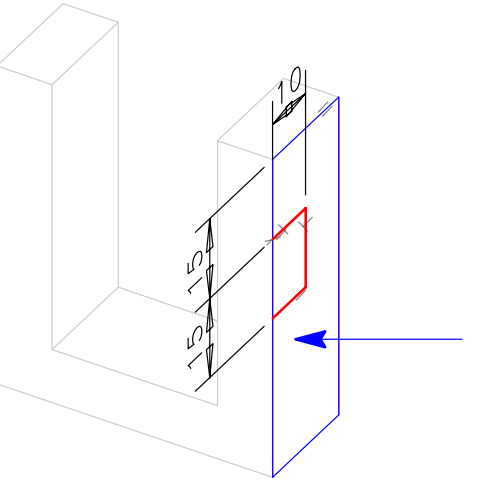
Update your part to turn off dimension display.

Remember

In the tutorial “Sketching and Constraining,” you used *Focus* to change the order of precedence for creating 2D constraints. In this section, you will use *Focus* to project a point to the 2D sketch plane.



Create a notch in one side by first sketching on the face as shown. Sketch only three lines that are parallel to the edges. Then add dimensions to the wireframe.



To define the section for the extrude, use *Stop at Intersections* and pick the segment of the face edge to close the sketch.

 *Section Options*

Section Options form

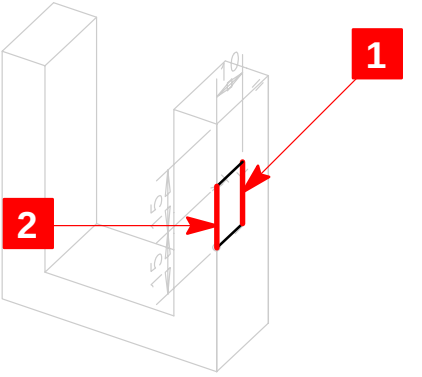
 *Stop at Intersections*

 *OK*

1

2

 (Done)

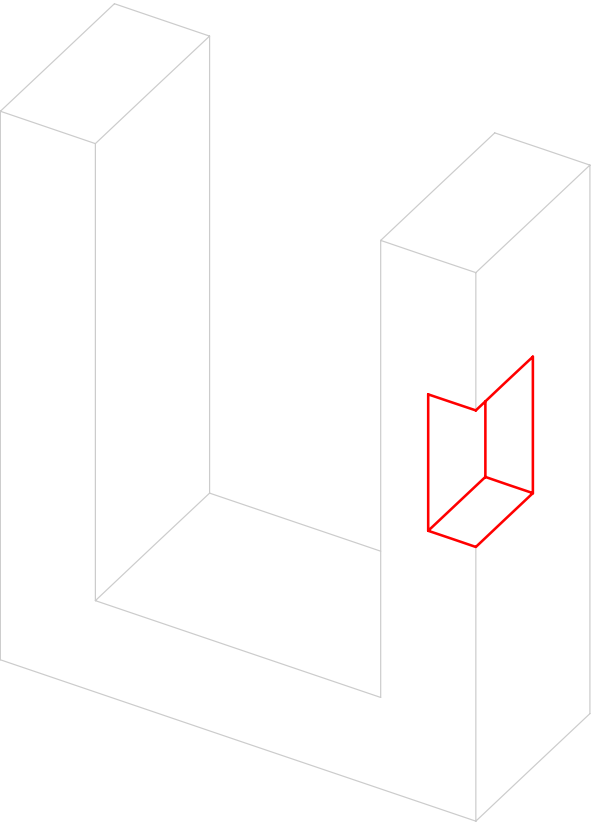


Extrude Section form

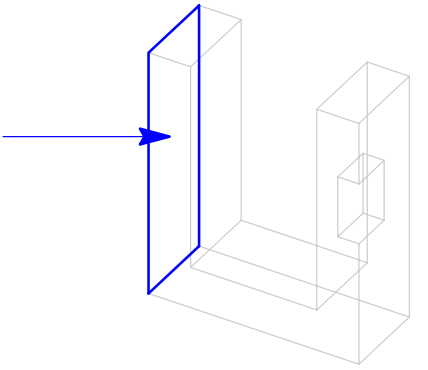
Cutout

Depth: 5

OK



Now, create a notch on the other side lined up exactly with the first cutout.



Before you sketch, focus on the existing cutout.

Focus

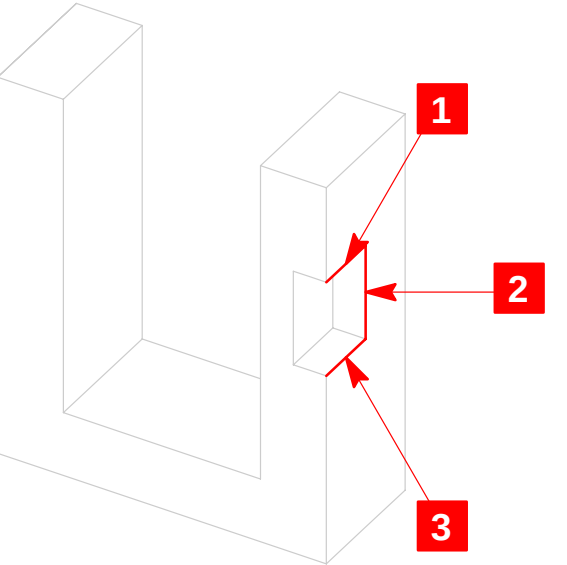
1

Focus

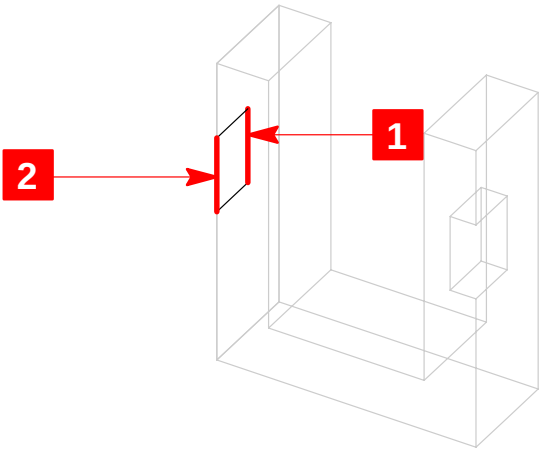
2

Focus

3



Extrude the cutout the same depth as the first one.



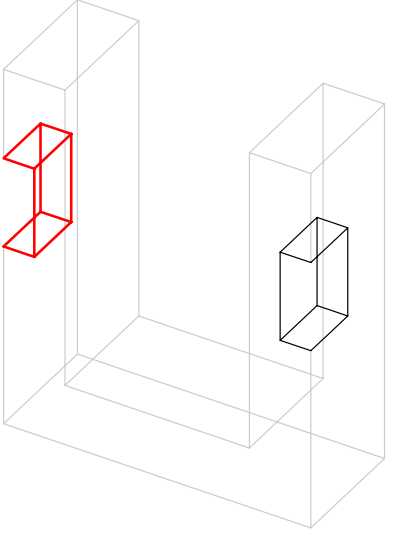
(Done)

Extrude Section form

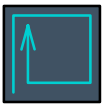
Cutout

Depth: 5

OK



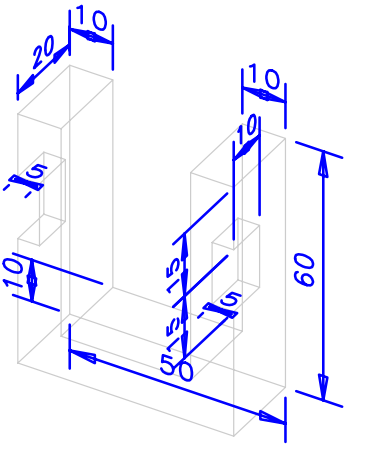
Display the dimensions.



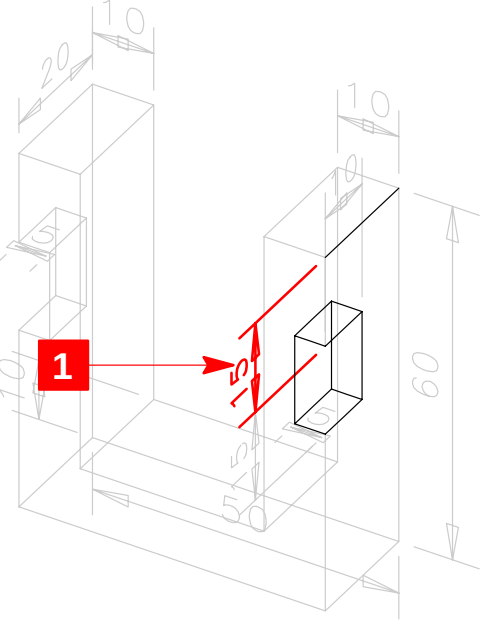
pick anywhere on part

(Accept)

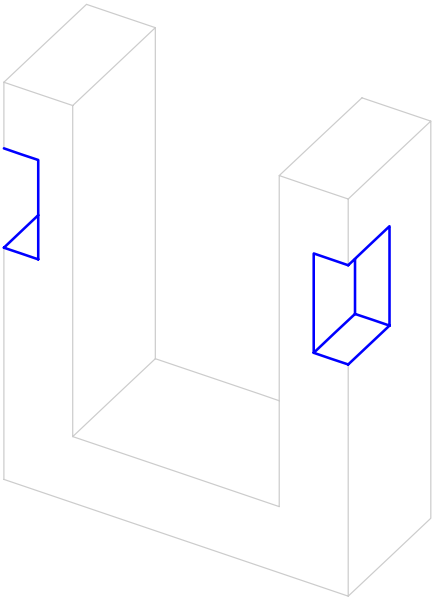
Show Dimensions



Then, modify the dimension that locates the notch from the edge of the part to 10 mm.



Update your changes.

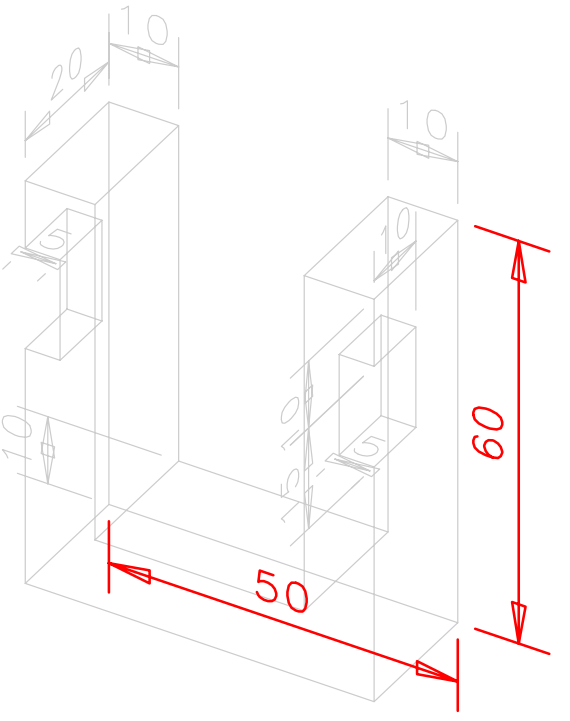


Things to notice

The position of both notches changes with the dimension because of the relationship between them.

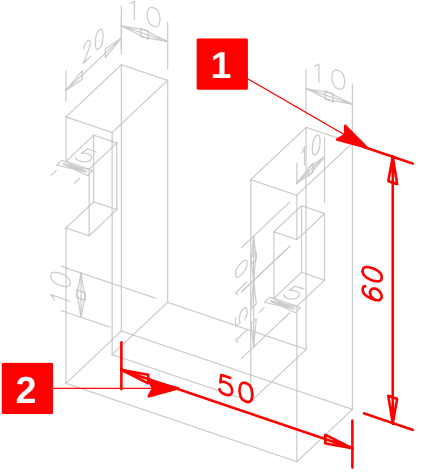
Create relationships by matching dimensions

Since each extrusion represents a different feature, you cannot constrain between their wireframe sketches directly. You can, however, match dimensions between the features.



Create relationships by matching dimensions

Show the dimensions again, then match the dimensions.



1

Modify Dimension

Match

2

OK

Modify the dimension to 55 mm.

2

Modify Dimension

55

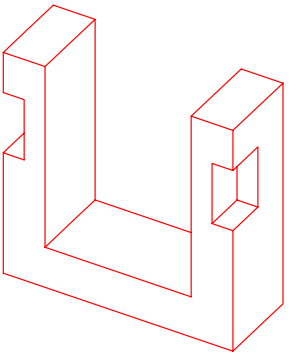
OK

Things to notice

Brackets appear around the dimensions that are matched. The dimensions update to reflect the changes to the dimension to which they're matched.

Create relationships by matching dimensions

Update your changes.



Recovery Point

File
Save

You can also match dimensions between features by creating and using equations.

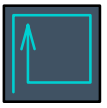
In the part you have on your workbench, assume that you want the slot between the arms to be centered and to be half the total width. This means that the width of the left and right arms must be the same.

One way to enforce this is to use equations, such as:

- slot width = total width / 2
- right width = left width
- left width + right width + slot width = total width

In the next steps, you will make the right width (RW), the left width (LW), and total width (TW) dimension names. Even though the slot width (SW) is not a dimension on the part, it can still participate in the equation.

Show the dimensions. Then modify the names, not the values, of the three dimensions shown.



1 pick dimension

Modify Dimension form

The name field is to the left of the **=** sign on the form.

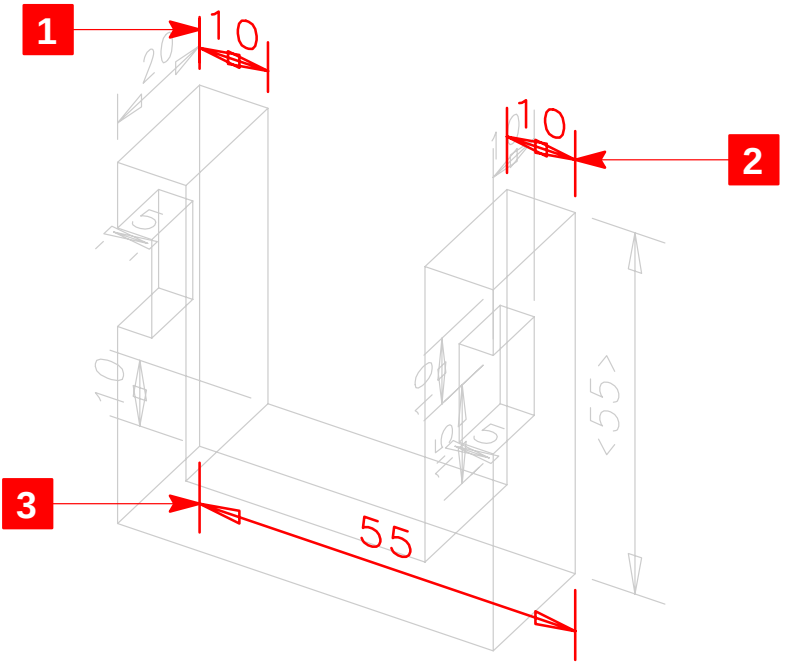
D# — change name to LW

OK

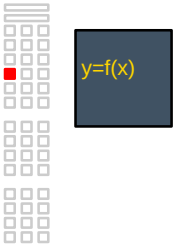
Repeat for the dimensions shown.

2 change name to RW

3 change name to TW



Enter equations to define the relationships to be solved.



pick anywhere on part

Equations form

Type in box at top of form:

LW=RW
SW=TW/2
LW+RW+SW=TW

Name table: LW

From Equation

Name table: RW

From Equation

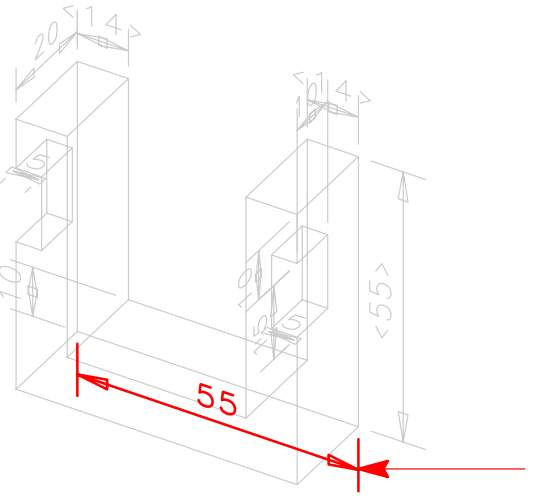
Apply

It is good practice to choose *Apply* before clicking *OK*. This way, you can see if you made any drastic mistakes before getting out of the form.

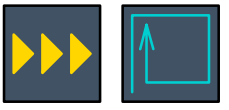
OK

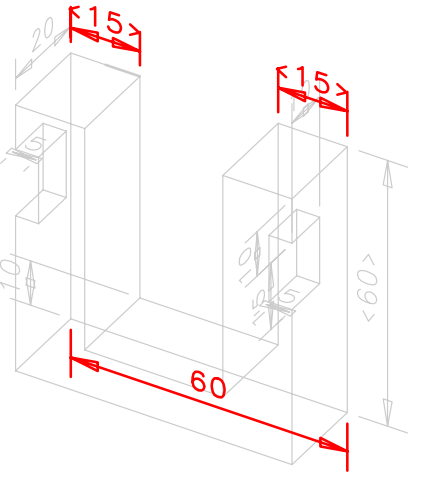
Although not required for this tutorial, if you use numeric constant terms in an equation, you can specify units (for, example: SW=20|mm|).

Modify the dimension indicated with the arrow to 60 mm.



Update the geometry and show the dimensions.





Things to notice

The equations used dimensions from multiple features and variables not on any feature. The software solved the algebraic equation to determine part dimensions.

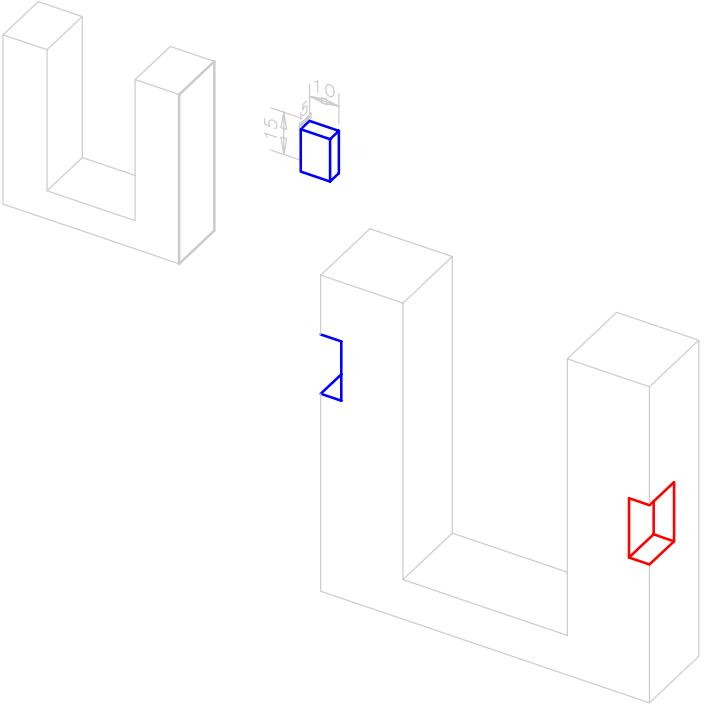
Recovery Point

File
Save

When you cut or join parts, you can store the relationship that defines how the features are positioned. The software stores this relationship with the part. If you modify the part, the feature will be repositioned using the same relationship.

You can specify a cut or join operation without defining a relationship. However, the results may not be what you expect when you modify dimensions later.

In the next steps, you'll suppress the notches in the part you created earlier. Then you'll create a new part that you'll use as a template for cutting new notches. Unlike when you used the *Focus* option, you use this technique to create identical features at different positions from each other on the part.

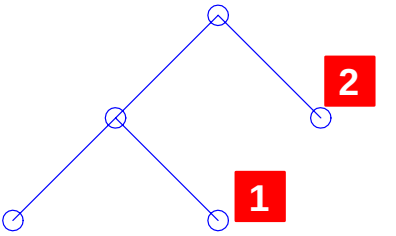


Suppress the notches on the part.

 pick anywhere on the part

 (Accept)

History Access form



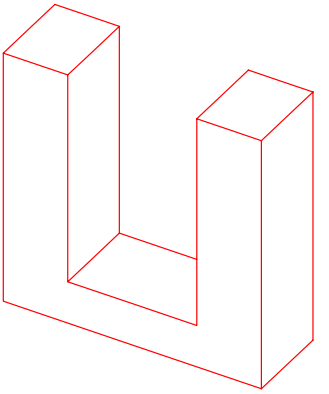
1 *Extrude*

2 *Extrude*

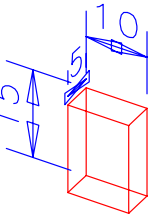
 *Dismiss*

Things to notice

Look at the history tree. Both features have an asterisk (*) at the end of the feature name. The asterisks indicate these features contain a face-to-face relationship with the surface on which they were sketched.



Create a part, 15 mm by 10 mm by 5 mm.

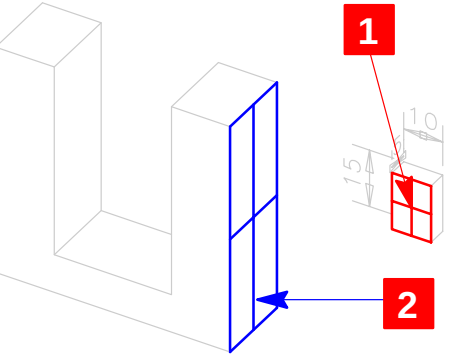
Name the part “cutter block.”

Cut the original part with the cutter block.

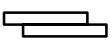
Verify the menu displays *Turn Relations OFF*. This indicates the relations are currently on. Otherwise, pick *Turn Relations On*.

1 *movable part*

2 *part to cut*



Locate the block along the edges of the part.



From Edges

3 *vertex*

4

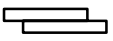
Check *I-DEAS Prompt*.

10

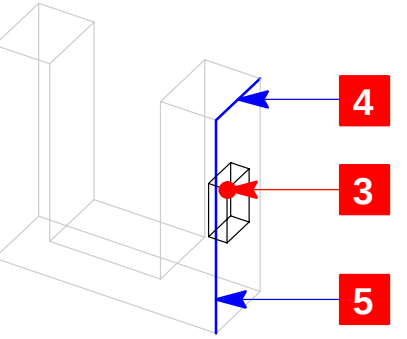
5

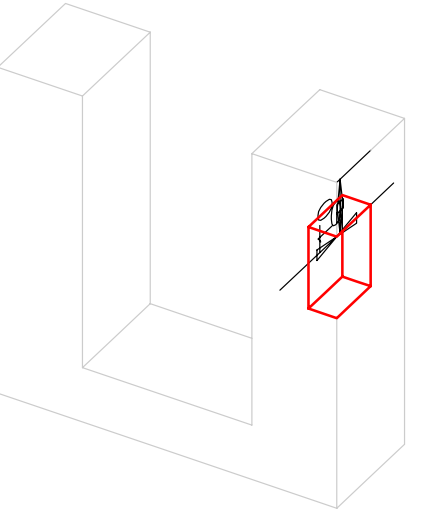
Check *I-DEAS Prompt*.

0



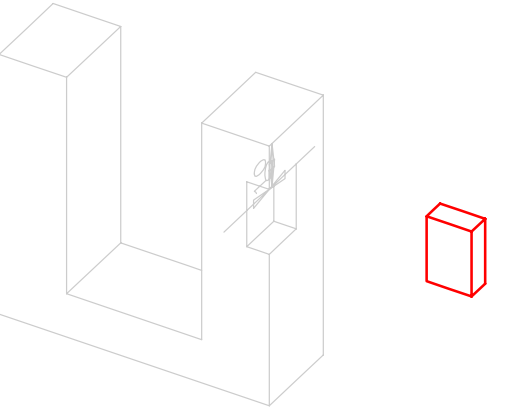
Done



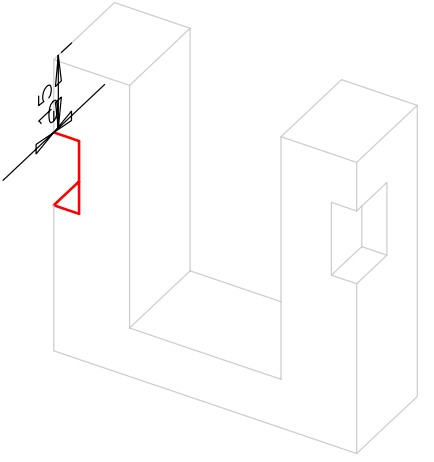


You will later see the implications of cutting features with the *Relations* switch turned OFF.

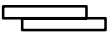
Get the cutter block from the bin. After you completed the cut in the previous step, the software put the cutter block part in the bin.



Cut to the other side of the part like you did in the previous step, but specify 15 mm from the upper edge and 0 mm from the side.



Recovery Point



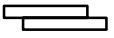
File
Save

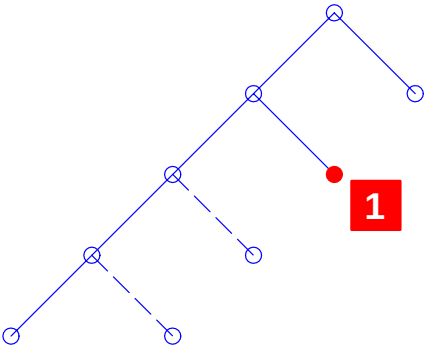
Delete the relations of one notch to the part.

 pick anywhere on the part

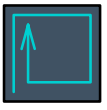
 (Accept)

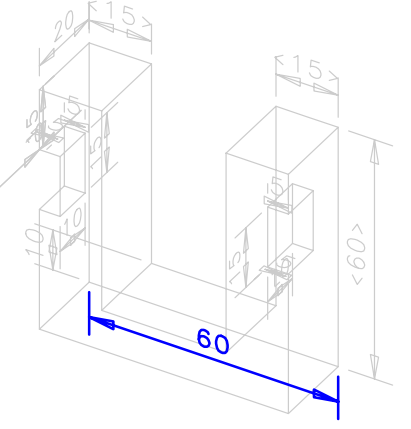
History Access form

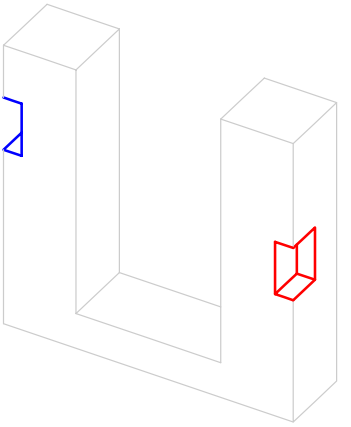
 *Delete Relations*



Modify the width of the part from 60 mm to 80 mm. Remember that you matched the height to the width, so the height will change too.







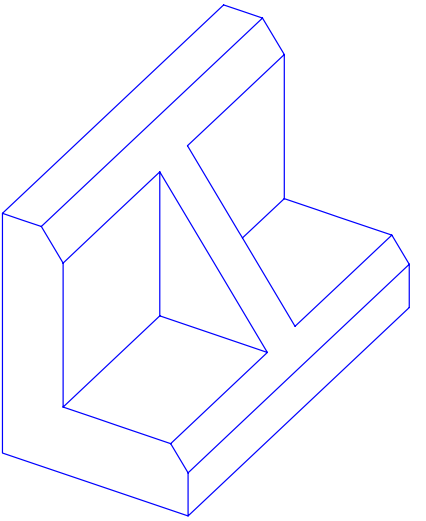
Things to notice

The notch without relationships doesn’t move. It didn’t maintain the distance from the upper edge after you changed the height of the part.

Delete or put away this part. It isn’t used in any other tutorials.

Create this part so the cutouts from either side line up and are cut to the same depth. If you modify the overall width of the part, the web should stay the same thickness and in the center.

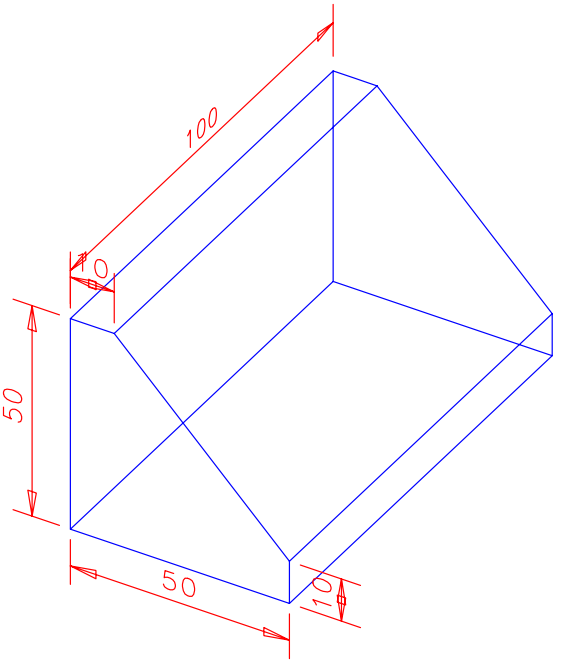
Either match dimensions or use equations.



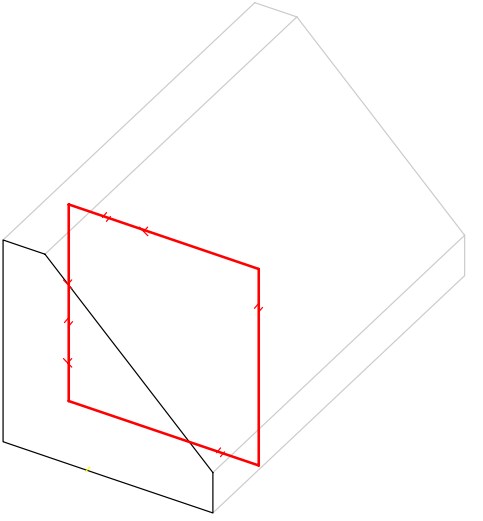
In equations, if you use a constant, such as SW=10, use SW=10|mm| to tell the software the units that apply to the constant.

Try this on your own. If you need help, refer to the next few pages, which give you hints on how to create the part.

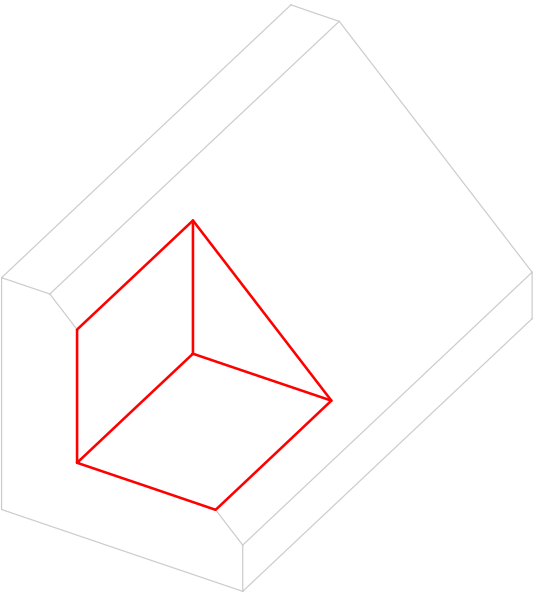
Step 1.



Step 2.

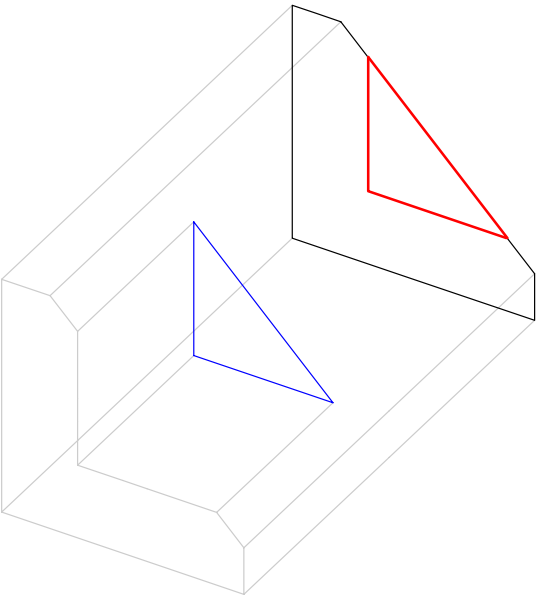


Step 3.

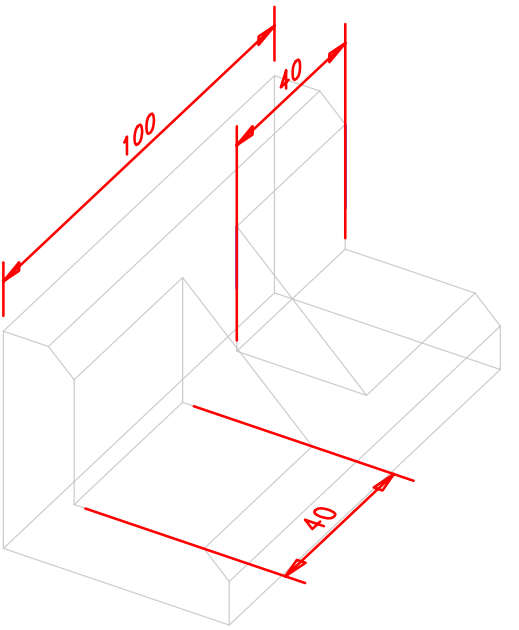


Step 4.

 *Focus*



Step 5. Use *Match* dimensions or equations to maintain a web thickness of 20 mm.



Hint

Using *Match*:

SW1 = (Total–20|mm|)/2

SW2 = (Total–20|mm|)/2

Using equations:

Web = 20|mm|

SW1 = SW2

SW1 + SW2 + Web = Total

(SW1, SW2 from equation)

Tutorial wrap-up

You have completed the Adding Features with Associativity tutorial.

Delete or put away any parts created in this tutorial. These parts are not used in any other tutorials.